



... for a brighter future

Quick Circuit 5000: Circuit Board Prototyping Part I

Geoff Waldschmidt



U.S. Department
of Energy

UChicago ►
Argonne_{LLC}



**Office of
Science**

U.S. DEPARTMENT OF ENERGY

A U.S. Department of Energy laboratory
managed by UChicago Argonne, LLC

Quick Circuit 5000

- Creates single or double sided circuit boards by etching traces from copper clad circuit boards and drilling holes for vias.
- Includes ISOPRO software for defining geometry and etching / drilling functions.
- ISOPRO interfaces with Quick Circuit 5000 hardware to create the circuit board.
- Quick Circuit precisely etches and drills the circuit board based on the geometry and drilling data supplied by ISOPRO.
- Vias must be added manually using bailbars after the circuit board is etched and drilled.

Steps for Circuit Board Software

- Software such as AutoCAD can be used to create the board geometry in .dxf file format.
- Convert .dxf file into a gerber (.ger) and a drill (.ncd) file representing the trace and via geometries, respectively.
- Import gerber and drill files into QuickCircuit prototyping software ISOPRO.
- 'Register' trace and drill files in ISOPRO to position the geometry, if necessary.
- Determine required tool sizes based on clearances on the circuit board.
- 'Isolate' the copper areas of the traces to prevent short circuits.
- 'Rubout' areas not removed by 'isolate' that should be copper-free.
- Create border around circuit board for final cut-out.
- Add text with descriptive name to document circuit board function.

ISOPRO: Software Definitions

Tools menu:
Defines geometry,
etching, drilling, and
routing operations

Drill file

Geometry file

Tools menu:

- Array
- Clearance Check
- Clip
- Create Arcs CW
- Create Arcs CCW
- Create Lines
- Create Pads
- Create Rectangles
- Create Text
- Create Vacuum Drill Array
- Insert Test Stamp
- Isolate Ctrl+I
- Measure
- New Tool
- Register Layers
- Reorder Entities
- Rubout
- Snap to Objects Ctrl+B
- Snap to Grid
- Database Tool
- Placement Tool
- Re-Sort all Polygons

Shape: Round Width: 62 Height: 0 mils mm

For Help, press F1

X: 6.07151 Y: 3.25631

Importing Files

- Geometry and drill files were originally created in Autocad as .dxf files and then converted into gerber and drill files.
- Gerber and drill files use simplistic compression technique (to reduce file size) called zero suppression.
 - All digits representing XY coordinates are not included in the gerber and drill files.
 - To recreate geometry, the number of integer and decimal digits, and the type of zero suppression must be specified.
 - The suppression parameters must be known to correctly import the geometry
- Geometries may also be created directly in ISOPRO using its built-in interface.

Gerber file (for a single trace in a coplanar waveguide)

```
%FSLAX55Y55*MOIN*%  
%ADD10R, 3.000000X1.000000*%  
%ADD11R, 3.000000X0.080000*%  
G75*  
%LPD*%G54D10*  
X150000Y50000D03*  
G54D10*  
X150000Y174000D03*  
G54D11*  
X150000Y112000D03*  
M02*
```

List of Aperture' sizes

'Aperture' used for geometry

XY Coords (assuming 5 integer and 5 decimal places with leading zeros)

Gerber File Format

Please Specify the following import settings

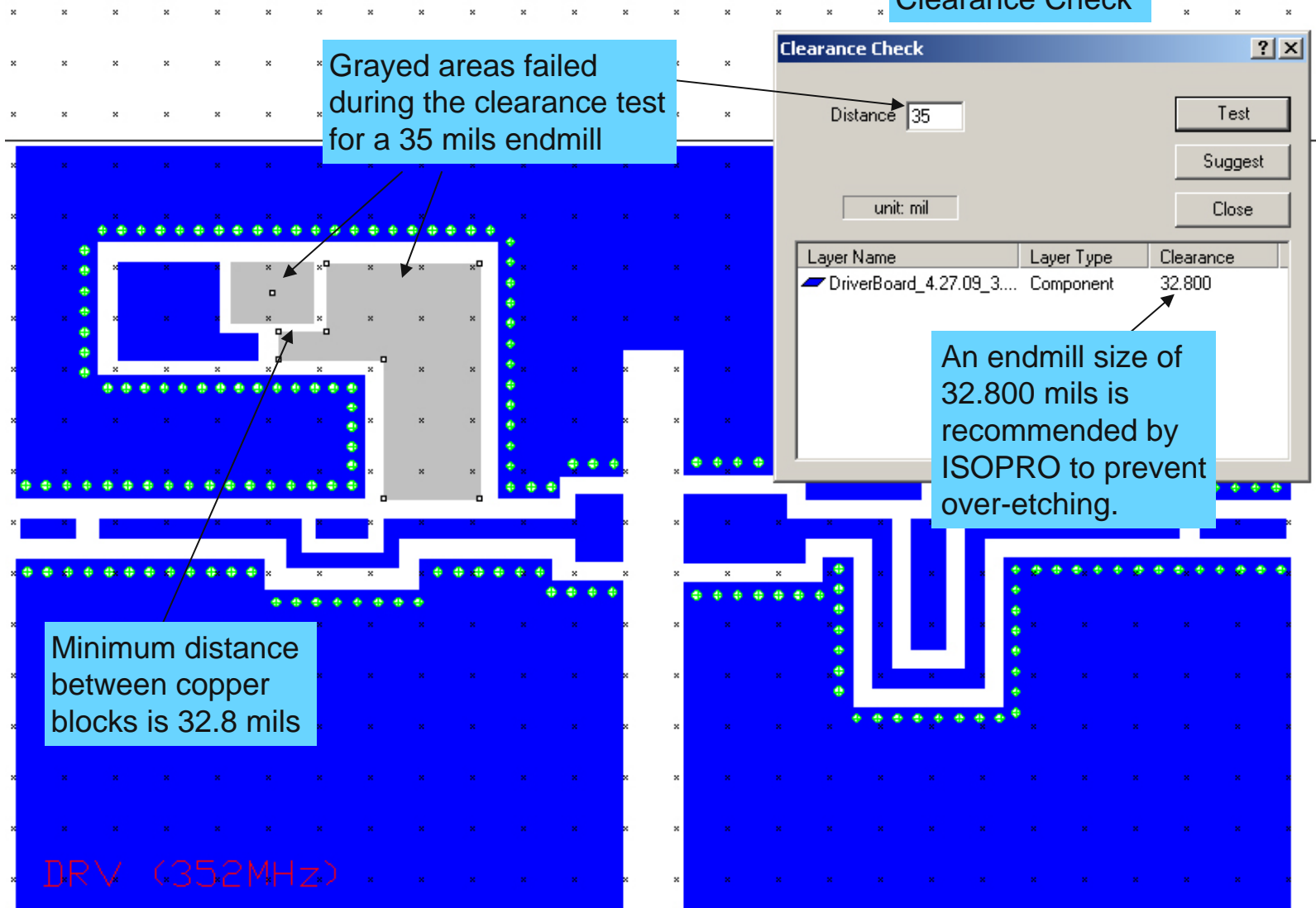
Digits
Integer Decimal

Mode
☒ Absolute ☐ Incremental

Zero Suppression
☒ Leading ☐ Trailing
☐ None ☐ Decimal

Units
☒ English ☐ Metric

Clearance Check



Isolate Layer Setup

Isolation prevents short circuits between traces

Menu: Tools > Isolate

The screenshot shows a PCB design interface. The main workspace displays a blue component layer with green traces. A dialog box titled "Isolate Layer(s)" is open in the foreground. The dialog has a table with two columns: "Name" and "Type". It lists two items: "DriverBoard_4.27.09_3..." of type "Component" and "DriverBoard_4.27.09.1..." of type "Drill". There are checkboxes for "expand pads" and "Remove redundant data". The "Remove redundant data" checkbox is checked. The dialog also has sections for "Pass 1", "Pass 2", and "Pass 3", each with a "Tool Size (inches)" field and an "expand pads" checkbox. The "Tool Size" fields are set to 0.02, 0.032, and 0 respectively. The "expand pads" checkboxes are unchecked. The "Isolate" and "Cancel" buttons are at the bottom right.

The component layer consisting of the traces will be isolated

Specifies 2 passes. The first uses a 20 mils endmill and the second uses a 32 mils endmill

Name	Type
DriverBoard_4.27.09_3...	Component
DriverBoard_4.27.09.1...	Drill

☒ Remove redundant data

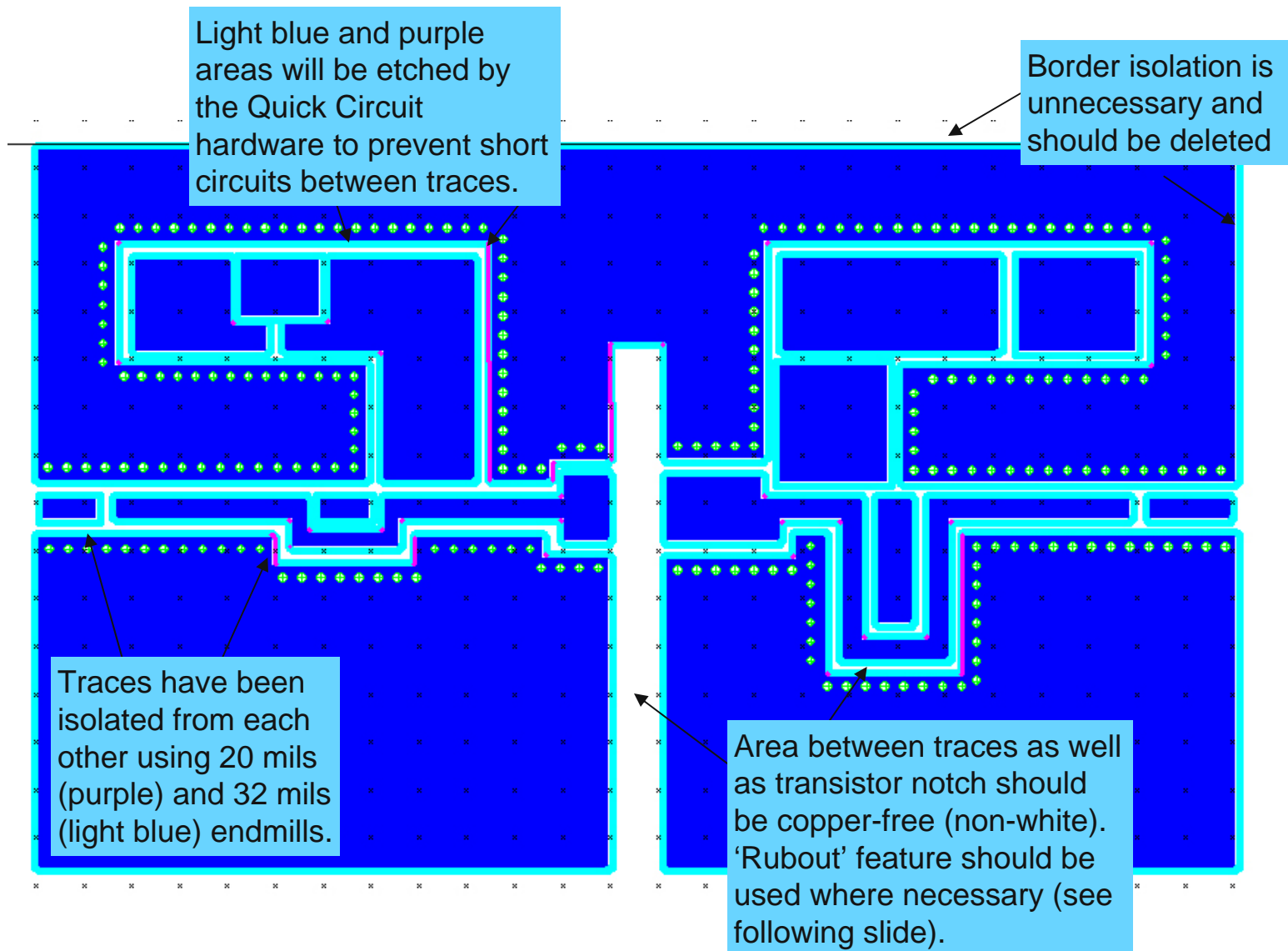
Pass 1
Tool Size (inches): 0.02
☐ expand pads

Pass 2
Tool Size (inches): 0.032
☐ expand pads

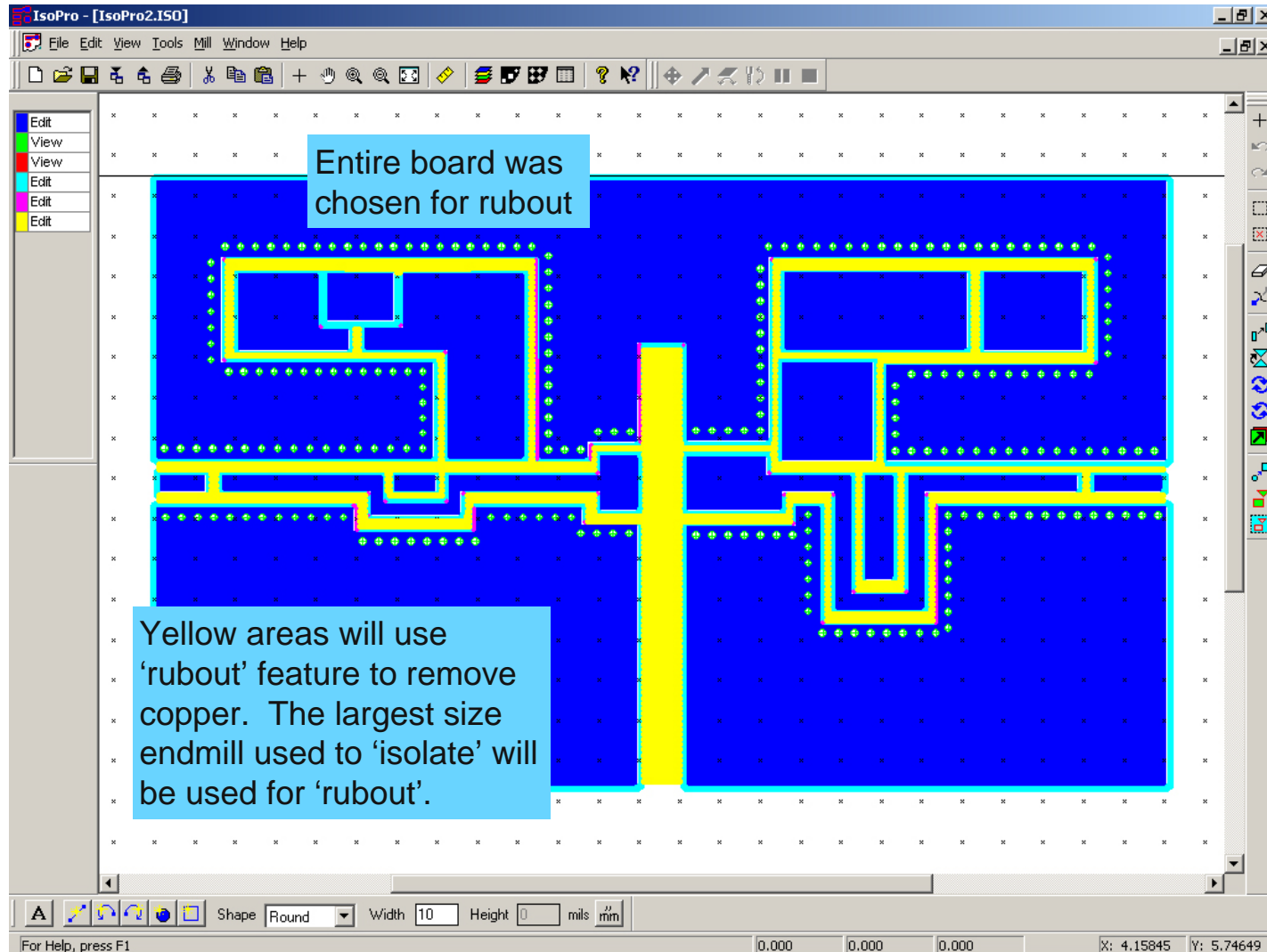
Pass 3
Tool Size (inches): 0
☐ expand pads

Isolate Cancel

Isolate Layer



Menu: Tools >
Rubout



Borders and Text

Borders and Text were created on their own layers so their construction can be uniquely defined when the Quick Circuit builds the prototype circuit.

Layers are described on the following slide

Text created using text tool

Border created using rectangle tool

Width was changed to 62 mils when border was created since that is the available routing bit size.

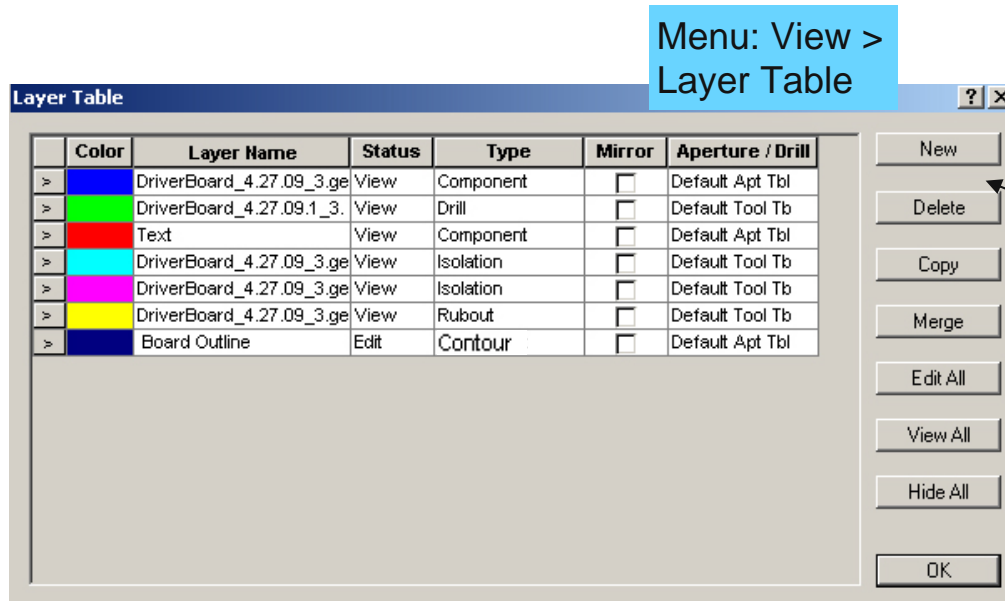
DRV (352MHz)

	Color	Layer Name	Status	Type	Mirror	Aperture / Drill
▼	Blue	DriverBoard_4.27.09_3.ge	View	Component	<input type="checkbox"/>	Default Apt Tbl
▼	Green	DriverBoard_4.27.09.1_3.	View	Drill	<input type="checkbox"/>	Default Tool Tb
▼	Red	Text	View	Component	<input type="checkbox"/>	Default Apt Tbl
▼	Cyan	DriverBoard_4.27.09_3.ge	View	Isolation	<input type="checkbox"/>	Default Tool Tb
▼	Magenta	DriverBoard_4.27.09_3.ge	View	Isolation	<input type="checkbox"/>	Default Tool Tb
▼	Yellow	DriverBoard_4.27.09_3.ge	View	Rubout	<input type="checkbox"/>	Default Tool Tb
▼	Dark Blue	Board Outline	Edit	Component	<input type="checkbox"/>	Default Apt Tbl

Shape Round Width 10 Height 0 mils mm

For Help, press F1

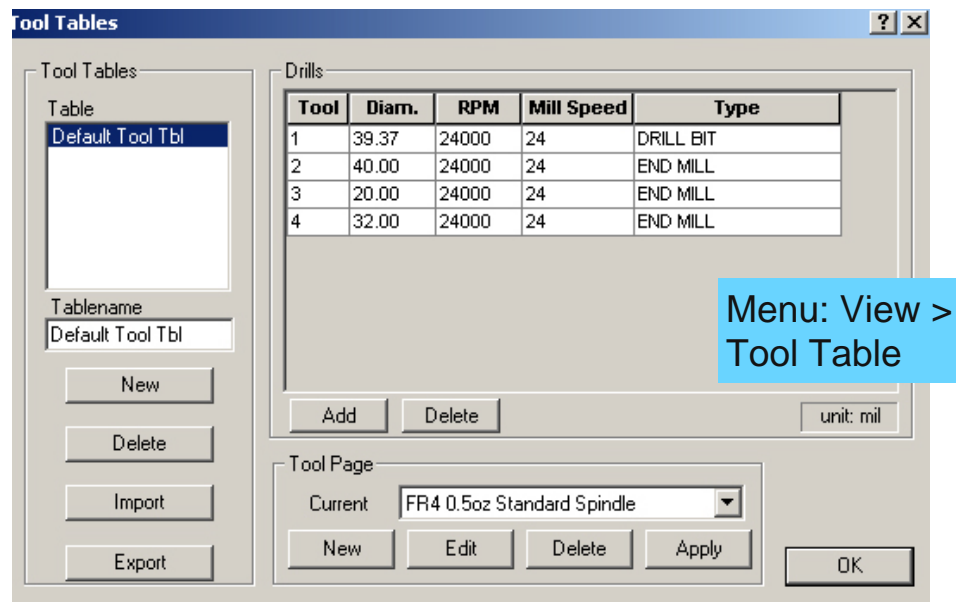
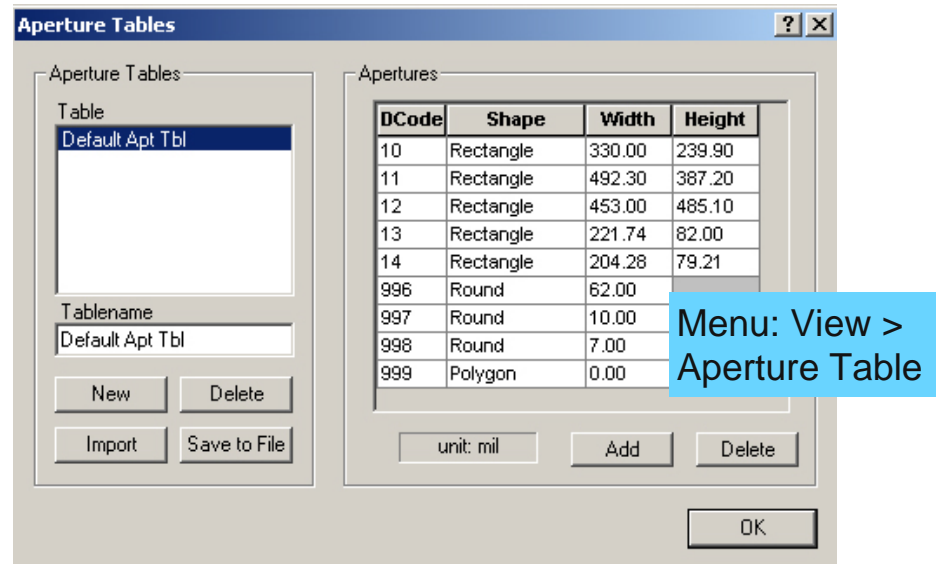
Layers



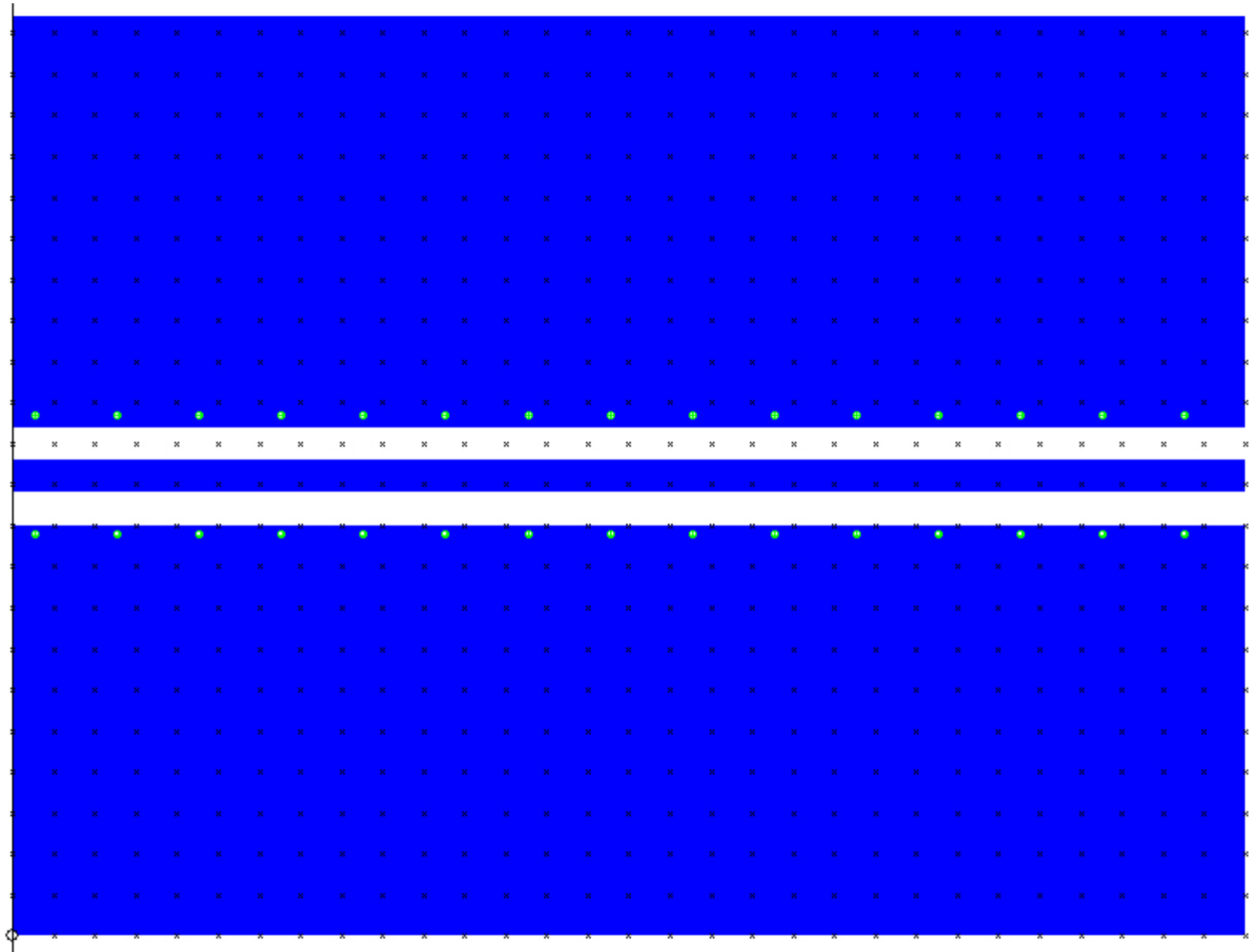
- Component layers represent the circuit board geometry and are used for defining isolation requirements.
- Drill layers define via locations.
- Isolation layers supply information on etching to the Quick Circuit 5000.
- Contour layers supply information on routing. Routing is used to cut the circuit board out of the bulk material.
- Rubout layers supply information on bulk copper removal.

Tool and Aperture Tables

- The tool table lists all endmills, drill bits, and routing bits that will be used in the prototyping process.
- The aperture table lists geometry data structures that are used in the translation of the gerber geometry file to a usable command list for the Quick Circuit hardware.
- The tool table should be inspected to ensure that the correct tools are required and that the tools are available for the Quick Circuit.



Example 1: Single Trace



Example 2: Partial Driver Board

